Australian Government

Department of Defence

Defence Science and
Technology Organisation

Fluent Code Simulation of Flow around a Naval Hull: the DTMB 5415

D.A. Jones and D.B. Clarke

Maritime Platforms Division
Defence Science and Technology Organisation

ABSTRACT

This report describes the application of the Fluent code to the numerical simulation of the free-surface flow around a model naval ship; the DTMB 5415. Simulations were performed using both a structured hexahedral mesh and an unstructured tetrahedral mesh of lower resolution. The results show that Fluent is able to accurately simulate the total ship resistance, near-field wave shapes, and the velocity field in the propeller plane. The results indicate that Fluent is a viable tool which could be considered for use in more demanding naval problems, such as the computation of ship wakes undergoing specified manoeuvres. A number of significant problems were encountered during the course of the work and have been summarized to facilitate future applications of the code to similar naval problems.

RELEASE LIMITATION

Approved for public release

APPROVED FOR PUBLIC RELEASE

Fluent Code Simulation of Flow around a Naval Hull: the DTMB 5415

Executive Summary

The DSTO has a requirement to calculate propeller loadings, performance and signatures for both naval surface ships and submarines. A detailed knowledge of the fluid velocity in the propeller plane is required in order to perform these calculations. Modern Computational Fluid Dynamics (CFD) computer codes which solve the Reynolds-Averaged Navier-Stokes (RANS) equations for complex geometries have been used to simulate viscous flow around ship hulls since the early 1990s.

This report describes the application of the Fluent code to the numerical simulation of the free-surface flow around a model naval ship; the DTMB 5415. The focus of the work is to study the capability of the Fluent code to accurately simulate the total ship resistance, near-field wave shapes, and the velocity field in the propeller plane.

Simulations were performed using both a structured hexahedral mesh and an unstructured tetrahedral mesh of lower resolution. Our results show that Fluent is able to accurately simulate the total ship resistance, near-field wave shapes, and the velocity field in the propeller plane. The total resistance coefficient calculated on the hexahedral mesh was found to agree with the experimental value to within 3.8%. The simulated wave shapes along the surface of the hull and in the near field computed on both the hexahedral and tetrahedral meshes were found to be in good qualitative agreement with the experimental profiles, with the degree of error from the results on the tetrahedral mesh being similar to those from the hexahedral mesh. The axial velocity component in the propeller plane was found to be the most difficult aspect of the flow to calculate accurately, although the maximum error was never more than 10%.

The results indicate that the Fluent code is a viable tool which could be considered for use in more demanding naval problems, such as the computation of ship wakes undergoing specified manoeuvres. A number of significant problems were encountered during the course of this work and they are described in the report to facilitate future applications of this code to similar naval problems.

Authors

David A. Jones
Maritime Platforms Division

Dr. David A. Jones obtained a B.Sc. (Hons) and Ph.D. in Theoretical Physics from Monash University in 1973 and 1976 respectively. He joined the then Materials Research Laboratories in 1983 after postdoctoral positions at the University of Strathclyde, Glasgow; Queen Mary College, London University, and the University of New South Wales, Sydney. During 1987/88 he was a visiting scientist at the Laboratory for Computational Physics and Fluid Dynamics at the Naval Research Laboratory, Washington, DC. He has authored 80 journal articles and technical reports and given more than 60 presentations at scientific meetings. His research has covered a variety of areas including polymer dynamics, the application of chaos theory to atomic and molecular physics, laser-plasma interaction theory, warhead design, air blast, detonation physics and computational fluid dynamics.

_____   _____

David B. Clarke
Maritime Platforms Division

David Clarke commenced work in the Maritime Operations Division (MOD) at DSTO in 1988 after completing a BSc at the University of Sydney. His work in MOD focused on magnetic sensors and instrumentation. He obtained a Graduate Diploma in Computer Engineering from RMIT in 1996. In 1998 he joined the Maritime Platforms Division to specialise in the hydrodynamics of underwater vehicles. His work on underwater vehicles encompasses experimental, empirical and computational hydrodynamics. The experimental work was conducted at the University of Tasmania's cavitation tunnel where he played a major role in developing the instrumentation and techniques used in this laboratory. In 2009 he completed a PhD at the University of Tasmania examining transcritical flow past ellipsoids. He is currently on a long term attachment to MARIN in the Netherlands after being awarded a Defence Science Fellowship.

_____   _____

Contents

1. Introduction

The DSTO has a requirement to calculate propeller loadings, performance and signatures for both naval surface ships and submarines. A detailed knowledge of the fluid velocity in the propeller plane is required in order to perform these calculations. Modern Computational Fluid Dynamics (CFD) computer codes which solve the Reynolds-Averaged Navier-Stokes (RANS) equations for complex geometries have been used to simulate viscous flow around ship hulls since the early 1990s. The first significant simulations were confined to tanker hulls and were relatively simple by today's standards; wave effects were absent, the free surface was considered as a plane of symmetry, and the most sophisticated turbulence model employed was the $k\text{-}\varepsilon$ model. A summary of these early simulations can be found in the proceedings of the SSPA-CTH-IIHR workshop on Ship Viscous Flow held in Gothenburg, Sweden, in 1990 [1].

As the speed of computers increased and more sophisticated RANS codes were developed more realistic simulations were able to be performed. These advances are well documented in the proceedings of several international conferences on the application of CFD techniques to ship flows which have been held every few years since 1990, most notably those in Tokyo in 1994 [2], Gothenburg in 2000 [3] and Tokyo in 2005 [4]. Significant information on the application of CFD codes to naval ships and submarines can also be found in recent proceedings of the "Symposium on Naval Hydrodynamics", which is a biennial symposium sponsored by the US Office of Naval Research (ONR) and was first held in 1956.

An example of the sophistication of current specialised CFD codes for the simulation of flow around naval hulls is provided by the work of Burg et al. [5] reported at the 24th Symposium on Naval Hydrodynamics. They used U^{2}NCLE, a three-dimensional unstructured, parallelized CFD code developed by the Computational Simulation and Design Centre at Mississippi State University to simulate the free surface flow around a fully-appended model of a US naval ship. The simulations produced by this code realistically captured the turbulent flow and vortices arising from the bulbous bow and the tips of the propulsors and rudders and were found to accurately model the rotation of the propulsors. Near-field wave shapes were accurately simulated using a nonlinear free surface algorithm.

CDFShip-Iowa, which was developed at The University of Iowa's Institute of Hydraulic Research (IIHR) under funding provided by ONR, is another example of a RANS code which has been specifically developed for surface-ship and marine-propulsor flow problems. Typical applications in naval hydrodynamics include the prediction of resistance (friction and pressure drag), wave profiles, sea keeping (the code is six-degree-of-freedom capable), and manoeuvring.

Wilson et al. [6] recently compared the performance of CFDShip-Iowa with that of two commercial CDF codes, Fluent, developed by Fluent Inc., and Comet, developed by CD-Adapco, in the prediction of ship generated wave fields. It was found that each of the codes had different advantages and disadvantages, and that each had certain specific requirements for obtaining accurate solutions of a surface ship wave field. It was also noted that the commercial solvers may have an advantage when complex hull shapes involving

appendages and propulsors are considered because they have the flexibility to use unstructured as well as hybrid meshes. Both Fluent and Comet also have the advantage of allowing solution based grid adaption techniques to provide finer grid resolution in restricted regions of the grid, such as in the air-water interface region, and hence may offer a more robust and computationally economical way to provide accurate free surface predictions in the vicinity of surface ship hulls.

The Hydrodynamics Group within MPD has considerable experience in the use of the Fluent code and its application to a variety of flow related problems in the naval context. Recent examples involving submarine related flows include the evaluation of hull modifications for the Collins Class Submarine [7,8], the analysis of flow disturbances created by fin-mounted cameras on a Collins Class submarine [9] and the simulation of the hydrodynamic forces on a model submarine [10]. More general applications have included the simulation of flow around a generic fin-body junction [11] as well as turbulence in the wake of a generic bluff body flow [12].

This report describes the application of the Fluent code to the numerical simulation of the free-surface flow around a model naval ship; the DTMB 5415. This model was conceived by the US David Taylor Model Basin (DTMB) in the early 1980s as a preliminary design for a surface combatant with a sonar dome bow and transom stern. The model has been studied extensively, both experimentally and numerically, and is one of the benchmark models for the ship hydrodynamics community, having been used for software validation at the Ship Hydrodynamics CFD Workshops in Gothenburg in 2000 [3] and Tokyo in 2005 [4]. Our focus in this work is to study the capability of the Fluent code to accurately simulate the total ship resistance, near-field wave shapes, and the velocity field in the propeller plane.

2. Computational Approaches to Free Surface Modelling

An important aspect of any simulation of the fluid flow around a surface ship is the method used to model the air/water interface. There are basically two approaches to this problem; surface fitting approaches or surface capturing approaches. In the surface fitting approach only the water side of the domain is simulated and the grid is adjusted to conform to the position of the free surface. In the surface capturing approach, the computational domain includes both the water and air, and the location of the interface on the mesh is computed from the solution of an auxiliary equation.

Most of the simulations in ship hydrodynamics prior to 2000 used surface fitting approaches [3]. Examples of this type of method include the simulations of Burg and Marcum [13], who used a nonlinear free surface algorithm in an unstructured finite volume RANS code to simulate flow around the DTMB 5415, as well as the work of Li [14], who used a nonlinear free surface algorithm on a structured mesh to model flow around a container ship, a tanker, and also the DTMB 5415. The advantage of these methods is that the air/water interface can be defined with great precision. There are significant disadvantages however. In time-dependent simulations with breaking waves the deformation of the interface is so great that changes in grid topology occur and these are difficult to handle with a surface fitting

approach. Even without a change in grid topology the deformation of the interface can be so great that strong cell deformation can occur and cause the computation to fail. Because of these disadvantages there has recently been an increase in the use of surface capturing techniques for the simulation of steady and unsteady ship motion in the field of naval hydrodynamics.

Surface capturing approaches can be classified as either Volume-of-Fluid methods (VOF) [15] or Level Set (LS) methods [16,17]. The VOF method can simulate the motion of two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fractions of each of the fluids throughout the domain. A volume fraction function is defined for each of the fluids in the simulation and is set to unity if the given fluid occupies a cell volume or is set to zero otherwise. When the interface between two fluids cuts through a computational cell the value of the function for a particular fluid represents the fraction of the cell volume occupied by that fluid. As the simulation evolves the volume fraction functions are convected by the underlying fluid flow. The VOF method therefore has the ability to treat quite complex interface evolutions and also has good mass conservation properties. The method is not ideally suited to problems involving surface tension effects however as problems can arise in constructing sufficiently accurate expressions for the surface curvature and surface normal vector from the volume fraction functions. This is not normally a problem in surface ship simulations however. Recent examples of the use of the VOF method to successfully simulate the flow around the DTMB 5415 include the work of Chen et al. [18] and Rhee and Skinner [19].

In the LS method the interface between the two fluids is represented by the set of all points of some higher order function for which the function has the value zero. This embedding function is then evolved with the flow and the zero level set at any instant signifies the location of the surface. Since a smooth function can be chosen to represent the initial surface location the calculation of the curvature and other geometric features of the surface as the flow evolves is relatively straightforward. One disadvantage of the method however is that it not good at conserving the mass of individual fluids on the mesh and this can lead to problems for lengthy time-dependent simulations.

In a two-phase LS method both air and water are simulated as a single fluid whose properties vary continuously across the free surface, whereas in a single-phase LS method the solution is computed only in the liquid phase. There has been significant development in the application of LS methods to naval hydrodynamic problems in the last few years. Yang et al. [20] have implemented a two-phase level set method with an immersed-boundary Cartesian grid method in version six of the CFDShip-Iowa code and used it to perform large-eddy simulations (LES) of several ship geometries, including the DTMB-5415, and promising results have been obtained. Di Mascio et al. [21] used a single-phase LS method to simulate both non-breaking and breaking flows around the DTMB 5415 and Carrica et al. [22] used a single-phase LS method in combination with a six degree of freedom (6DOF) algorithm and dynamic overset grids to calculate the sinkage and trim and pitch and heave motions for the DTMB 5512, which is a smaller scale model of the DTMB 5415.

There are two approaches available for the numerical simulation of multiphase flows in the Fluent code; the Euler-Lagrange approach and the Euler-Euler approach. The Euler-Lagrange

approach is essentially a Lagrangian discrete phase model in which the fluid phase is treated as a continuum by solving the Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. This is obviously relevant to particle-laden flows and is inappropriate for the sea/air interface problem considered here, although it may become relevant in the future if breaking waves need to be considered. An example of this type of application would occur during the simulation of bubbly ship wakes for the calculation of torpedo tracking signatures.

In the Euler-Euler approach the different phases are treated as continua and three different Euler-Euler multiphase models are available; the mixture model, the Eulerian model, the volume of fluid (VOF) model. The mixture model is designed for two or more phases in which one of the phases represents a particulate. It solves for the mixture momentum equation and then prescribes relative velocities to describe the dispersed phases. Applications of the mixture model include particle-laden flows, sedimentation, and bubbly flows. The Eulerian model is the most complex of the multiphase models in that it solves a set of momentum and continuity equations for each phase and coupling is achieved through pressure and interphase exchange coefficients which depend upon the type of phases involved. This model is suitable for the simulation of bubble columns, particle suspensions and fluidized beds.

The VOF model is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. A single set of momentum equations is shared by the fluids and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. This model is ideally suited to applications involving free-surface flows, filling, sloshing , the motion of large bubbles in a liquid, and the motion of liquid after a dam break.

3. Previous CFD studies of flow around the DTMB 5415

Figure 1. Grey scale image of the DTMB model 5415 geometry

The DTMB model 5415 was first studied extensively using CFD techniques when it was chosen to be one of three test cases in the Gothenburg 2000 workshop for computational fluid dynamics applied to ship flows. Participants were asked to focus in particular on the total resistance, the wave profile along the hull and at selected locations close to the ship, the overall wave pattern in the near field and the mean flow velocities near the stern, particularly at the propeller plane. Experimental data for this model has been obtained by three laboratories; DTMB, IIHR, and INSEAN (Instituto Nazionale per Studi ed Esperienze di

Architettura Navale). A detailed comparison of this data is provided in the paper by Stern et al.[23]. Figure 1 shows a gray-scale image of the model geometry and the test conditions for the simulation are given in Table 1.

Seven CFD codes were used to model the DTMB 5415 at the Gothenburg 2000 workshop; CFX, COMET, FINFLO, ICARE, CFDSHIP, MGSHIP and UNCLE. The main conclusions drawn from the analyses of the seven simulations and a comparison with the experimental data were as follows:

1. The computed total drag coefficient (C_T) differed from the experimentally measured value by a maximum of $\pm 7\%$, and the average value over the seven codes was almost exactly the same as the data (0.5% higher).

2. Most of the codes had difficulty accurately simulating the wave profiles in the near field. The main problem was the under prediction of the peak in the bow wave due to excessive numerical damping.

3. The flow in the propeller plane was reasonably well predicted by half of the codes. No vortex could be detected in the cross-flow vector plots, but detailed vorticity plots were able to capture the vortex shed from the bulbous bow.

Table 1. Test Conditions for scale model at 20 knots

Scale Ratio	24.832
Length (L)	5.72 m
Draft (T)	0.248 m
Wet Surface Area (S)	4.861 m^2
Advance Velocity (U_0)	2.0637 m/s
Froude Number (Fn)	0.2755 m/s
Reynolds Number (Re)	1.26×10^7
Sinkage at FP	-0.0028L
Sinkage at AP	-0.0009L

With regard to the flow into the propeller plane for the DTBMB 5415, Gorski [24] has commented "*The dominant flow feature is a vortex created at the sonar dome that flows downstream to the propeller plane. In general… RANS computations with two-equation models can predict this vortical flow well. The exact position and strength of the vortex, which depends on grid resolution and turbulence modelling, are not necessarily obtained, but enough detail of the average inflow and wake deficits for propeller design are obtained. At full scale the vortex will be closer to the hull and may or may not impact the propeller significantly.*"

Subsequent to the Gothenburg 2000 workshop the DTMB model 5415 has been used as a test case for a number of other CFD codes and interface modelling techniques. Rhee and Skinner [19] used Fluent version 6.2 to simulate flow around the DTMB 5415 under the test conditions shown in Table 1. They used a half domain hexahedral cell mesh containing approximately

1.8 million cells and the HRIC (High Resolution Interface Capturing) VOF formulation within Fluent to capture the air/water interface. Simulations were performed using two different turbulence models; the shear-stress-transport k-ω model of Menter and the Reynolds stress transport model. The total resistance coefficient was under predicted by 9.4% but the wave profiles along the hull surface and in the near field agreed very well with the measured data. Peak values were under predicted away from the hull however and this was attributed to numerical diffusion enhanced by stretched cells in the area outside of the imminent neighbourhood of the hull. The velocity components in the propeller plane showed good agreement with the experimental values.

Wood et al. [25] used the finite volume commercial solver CFX to perform a similar calculation for the DTMB 5415. The test conditions were again the same as those shown in Table 1. They used the VOF model implemented in CFX to model the air/water interface and the shear-stress-transport k-ω model for the turbulent flow. Simulations were performed using both structured hexahedral meshes on a half domain mesh or unstructured tetrahedral meshes with prism boundary layers on the free surface in an attempt to obtain accurate wave profile simulations. Results calculated on the unstructured meshes however showed very poor agreement with the experimental results. On the structured hexahedral mesh they calculated the drag coefficient to have the value 4.368×10^{-3}, which is 3.2% higher than the experimental value of 4.23×10^{-3} [3]. Wave profiles along the hull and in the near field were also obtained and in general showed good agreement with the experimental results, although their calculated profiles also showed a small loss in wave amplitude in the peaks of the profiles, which they attributed as being due to numerical diffusion. Axial velocity plots in the propeller plane also showed good agreement with the experimental vales.

Di Mascio et al. [15] have used the DTMB 5415 to validate their single-phase level set method coupled with a standard, in-house, CFD RANS solver. Their code uses the finite volume technique with pressure and velocity co-located at the cell centre and a second order Godunov type algorithm to discretize the RANS equations. The Spalart-Allmaras one-equation turbulence model was used for the turbulent flow. The physical domain was meshed using a structured hexahedral grid with approximately 2.5 million cells. For the test conditions given in Table 1 they calculated a total drag coefficient of 4.39×10^{-3}, which is 3.8% higher than the experimental value. Their computed wave shape along the hull shows excellent agreement with the experimental data, although there is a slight reduction in peak value at the bow compared to the experimental vale, which Di Mascio et al. propose can be explained by the way in which the photographic and digitizing system was used to obtain the experimental results.

4. The Fluent Code

Fluent version 6.3 is a cell centred finite volume general purpose CFD computer code developed by Fluent, Inc. and now marketed by ANSYS, Inc. an engineering simulation software provider which acquired Fluent, Inc. in 2006. The Hydrodynamics Research Group within MPD has been using this code for several years to simulate a number of problems

involving submarine related flows [7,8,9,10]. This report describes our first application of the code to surface ship flows.

A distinct difference between the simulations described here and those reported earlier is the presence of the air/sea interface in the calculation. As explained in Section 2, there are two main approaches to maintaining a distinct interface in a simulation, either surface fitting approaches or surface capturing approaches. Fluent has implemented the surface capturing approach by using the VOF scheme for general multiphase flow modelling. This involves defining a volume fraction function for each of the fluids throughout the domain and then convecting the volume fraction of each fluid with the average fluid flow. The interface between the two fluids is then reconstructed from the volume fraction function for each of the fluids in the vicinity of the interface.

Fluent 6.3 offers a choice of several different methods for interface reconstruction. The appropriate scheme to use depends on whether the simulation is steady-state or time-dependent and whether an implicit or explicit time discretization scheme is used. The results described in this report were calculated from implicit steady-state simulations and a High Resolution Interface Capturing (HRIC) scheme was used to ensure a sharp interface between the two fluids. This is a modification of the HRIC scheme described by Muzaferija et al. [26]. It consists of a non-linear blend of upwind and downwind discretizations so that the computed fluxes of the volume fractions do not underflow or overflow the cells. Standard interpolation schemes are used to obtain the face fluxes whenever a cell is completely filled with one fluid.

Simulating the motion of a surface ship not only requires maintaining a sharp interface between the sea and air phases but also requires specification of appropriate boundary conditions at the inlet and outlet of the domain for each of the fluids in the simulation. One way to do this is to write a User-Defined Function to specify the total pressure and VOF profile for each of the fluids at both the inlet and outlet. The other option is to use the new Open Channel Boundary Condition implemented in Fluent 6.3. Using this method the inflow/outflow condition is specified via the inflow velocity and the free surface level and the pressure and VOF profiles are automatically calculated. This simplifies the initial set-up of the simulation and was the boundary condition used in the simulations described here.

Fluent 6.3 offers a variety of different RANS turbulence models. These include the Spalart-Allmaras model, the k-ε model, and the k-ω model, all of which are based on the Boussinesq approach, which assumes that the calculated turbulent viscosity coefficient is isotropic. However for complex flows involving streamline curvature, swirl, rotation, and rapid changes in strain rate this is not the case. For these flows Fluent provides the Reynolds Stress Model (RSM), which solves the RANS equations by solving additional transport equations for each of the individual Reynolds stresses. This means that five additional transport equations must be solved in three-dimensional flows and the simulation time increases substantially. Since the simulations described in this report involve only straight-ahead motion at minimal angles of attack we expect the flow to remain largely attached to the hull, hence we have employed the standard k-ε turbulence model and standard wall functions for these simulations. Rhee and Skinner [19] have previously used both Menter's shear-stress transport model and an RSM model to simulate the same test case as that considered here and have shown that the

boundary layer remains attached to the hull for the majority of the flow and is similar to an equilibrium boundary layer flow.

5. Simulation Results with a Fully Hexahedral Mesh

 A fully structured hexahedral mesh containing approximately 3.8 million cells was constructed using the Fluent preprocessor, Gambit. As the flow has a plane of symmetry about the centre plane a half domain grid was used. The hull surface was first meshed using quadrilateral elements. A volume forward of the bow was then created by projecting the outline of the bow surface towards the inlet. The mesh for this volume was created from the projection of the surface mesh on the matching portion of the bow. Projection of the stern geometry and its surface mesh towards the outlet created the corresponding volume and mesh for the stern. The initial section of the bow projection was angled to minimise the skew of the elements. This was unnecessary for the stern geometry as the orientation of the stern is approximately parallel to that of the outlet and thus there is no tendency for highly skewed elements to occur in this region.

The deck of the vessel and the upper surface of the volumes projected from the bow and stern were in turn projected to the desired height for the top of the fluid domain, thus creating the volumes above these surfaces. The mesh for these volumes was created from the projection of these surface meshes. A "C" volume and grid in the vertical plane was wrapped around the hull and projected volumes. The grading of the elements around the bow was adjusted to minimise the skewness around the sonar dome. Care was taken to ensure a suitable density of elements at the waterline, bow, stern, and wake region. Details of the surface mesh over the bow are shown in Figure 2, and the surface mesh over the entire hull is shown in Figure 3.

The experimental conditions for the simulation were the same as those in the Gothenburg 2000 workshop and are shown in Table 1. The attitude of the model with respect to the coordinate axes was set according to the experimentally measured sinkage and trim values before meshing commenced. The origin of the coordinate system was located at the midship intersection of the calm water free-surface and the centre plane. The open channel boundary condition was used to specify the inlet and outlet boundary condition, a symmetry plane was used along the centre plane, and the remaining boundary surfaces along the exterior of the domain were set to slip wall conditions. The longitudinal extent of the mesh was $-2Lpp \leq x \leq 5Lpp$ and the radial extent was $0.0 \leq r \leq 2.5Lpp$. (Lpp denotes the length between perpendiculars).
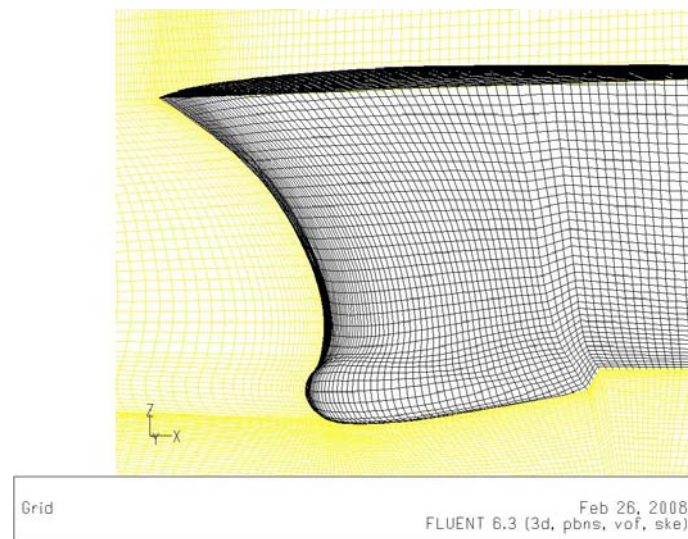
Figure 2. Detailed view of the surface mesh on the bulbous bow

An implicit steady-state cell based solution procedure was used to solve the equations. The SIMPLE algorithm was used for the pressure-velocity coupling, the PRESTO scheme for the pressure interpolation, the 2nd order upwind scheme for the solution of the momentum equations and the modified HRIC scheme for the solution of the volume fraction equations. The relaxation factors were typically set to 0.2. The standard k-ε turbulence model with equilibrium wall functions was used to simulate turbulent flow. The y^+ values for the wall adjacent cells over the hull were in the range $80.0 \leq y^+ \leq 100.0$, which is comfortably within the guidelines for the use of the wall function approach.

Initial convergence of the simulation was found to be considerably enhanced by paying careful attention to the initialization procedure. The primary phase should always be set to the lower density fluid, which in this case is air. The specified operating density should be set to that of the primary phase and the reference pressure location should be set to a region which will always contain the primary phase. It was also found helpful to initialize the entire water domain with the correct hydrostatic pressure profile and to initialize both the water and air domain to the same velocity.
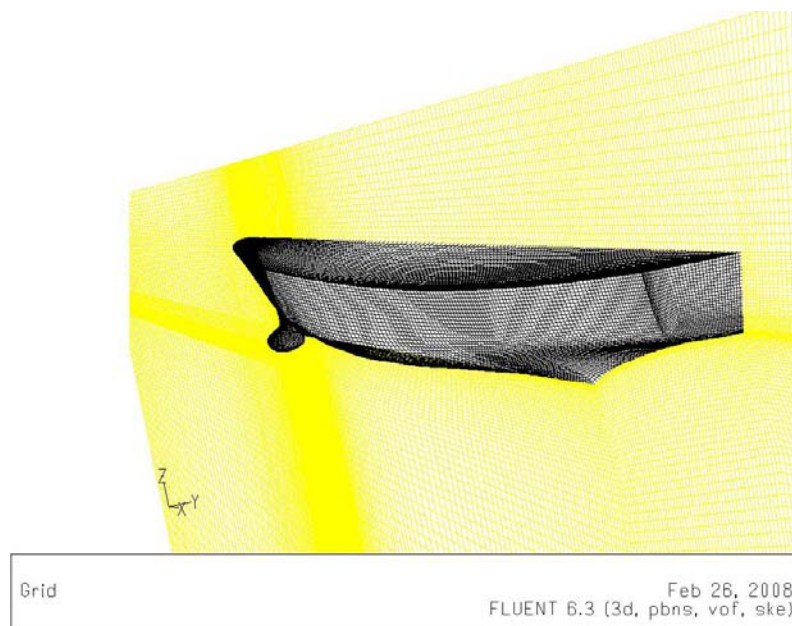
Figure 3. Surface mesh over the entire hull

Initial simulations using this mesh were discouraging. After 13,000 iterations the water surface showed no signs of developing the classical Kelvin wake pattern, even though all residuals had decreased by several orders of magnitude at this stage. More importantly, the water surface contours indicated the presence of significant computational errors occurring at the inlet and at the far field boundary. The source of these errors was not definitively identified but it was thought that they originated either from an incompatibility of the Open Channel boundary condition with the curved outer boundary surface, or from the irregular nature of the surface mesh over the inlet boundary due to the meshing constraints imposed by the projection of the bulbous shape of the lower bow. The nature of these errors is indicated in Figure 4, which shows a contour plot of the vertical height of the water surface, which is the iso-surface of all cells where the volume fraction of water has the value 0.5. The curved nature of the far field geometry is also evident in Figure 4, which shows the surface mesh over the upstream inlet boundary. The physical extent of the domain is evident from the outline of the hull.

As a first attempt to obtain a realistic Kelvin wake pattern it was decided to increase the mesh resolution in the vicinity of the water surface. To do this the "adaption" feature of the software was used, which automatically refines the mesh in a given region by splitting each hexahedral cell into eight smaller hexahedral cells. Whilst being easy to implement, this procedure results in a significant increase in the total number of cells on the mesh. An adaption was then performed in the region of the mesh 20 cm above and below the water line and this took the total number of cells to approximately 9.5 million. The simulation was then allowed to run out to 35,000 iterations but the results were again discouraging as no improvement was seen in the shape of the water surface.
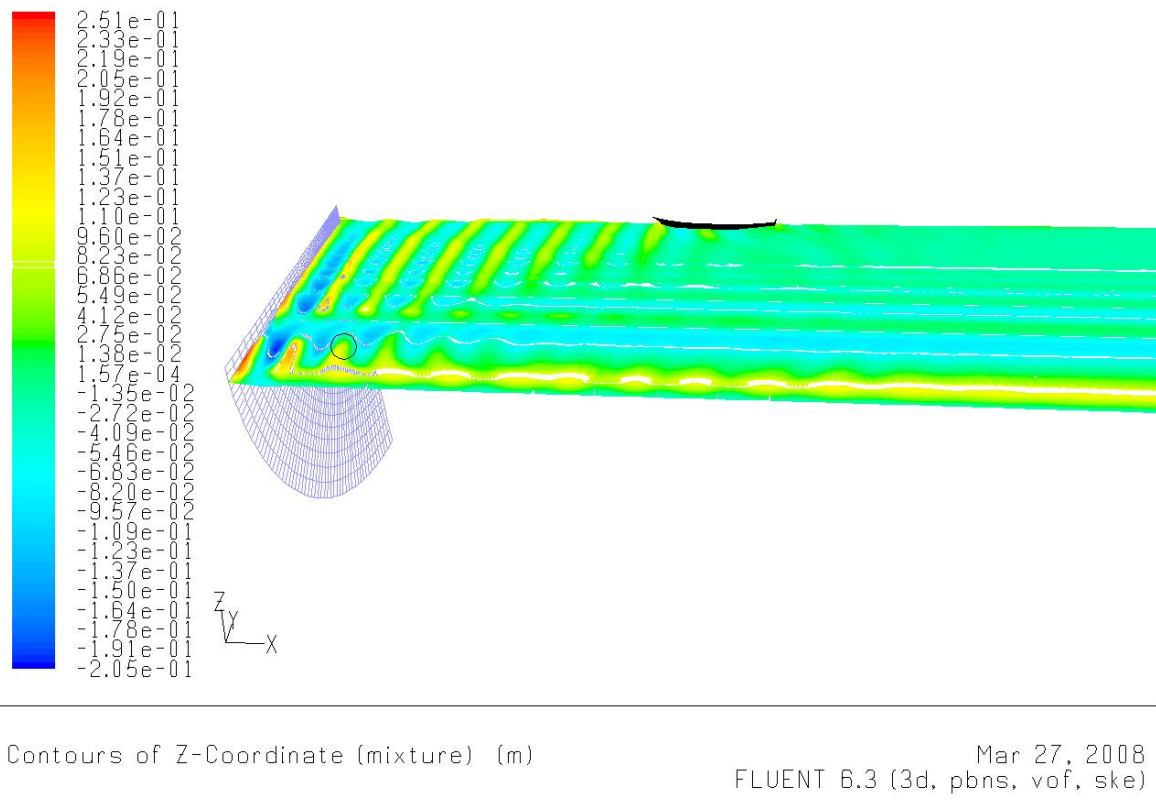
Contours of Z-Coordinate (mixture) (m)   Mar 27, 2008
                                                FLUENT 6.3 (3d, pbns, vof, ske)

Figure 4. Water surface elevation and nature of curved outer boundary

A realistic Kelvin wake pattern was finally obtained by addressing the problems described in the previous paragraph. Two changes were made: (a) the curved outer boundary surface was removed and replaced by a rectangular box geometry. The resulting volume was then meshed using regular hexahedral cells. (b) Additional hexahedral cells were added upstream of the existing mesh to provide a uniform surface mesh for the imposition of the inflow boundary conditions. A non-conformal interface was used between the face of this additional mesh and the irregular surface mesh at the original mesh boundary. These changes increased the number of cells in the mesh to approximately 5.6 million. The new mesh was then run out to 8,000 iterations until the calculation converged. It should be noted that our criterion for convergence is based on a combination of monitored metrics and a reduction in calculated residuals. In the simulations reported here we monitored the shape of the free surface as well as the reduction in residuals. We considered the calculation to have converged when an increase in the number of iterations produced no further change to the shape of the free surface and left the magnitude of the residuals unchanged. The resulting wave pattern is shown in Figure 5, and Figure 6 shows a more detailed view of the wave pattern in the vicinity of the hull.
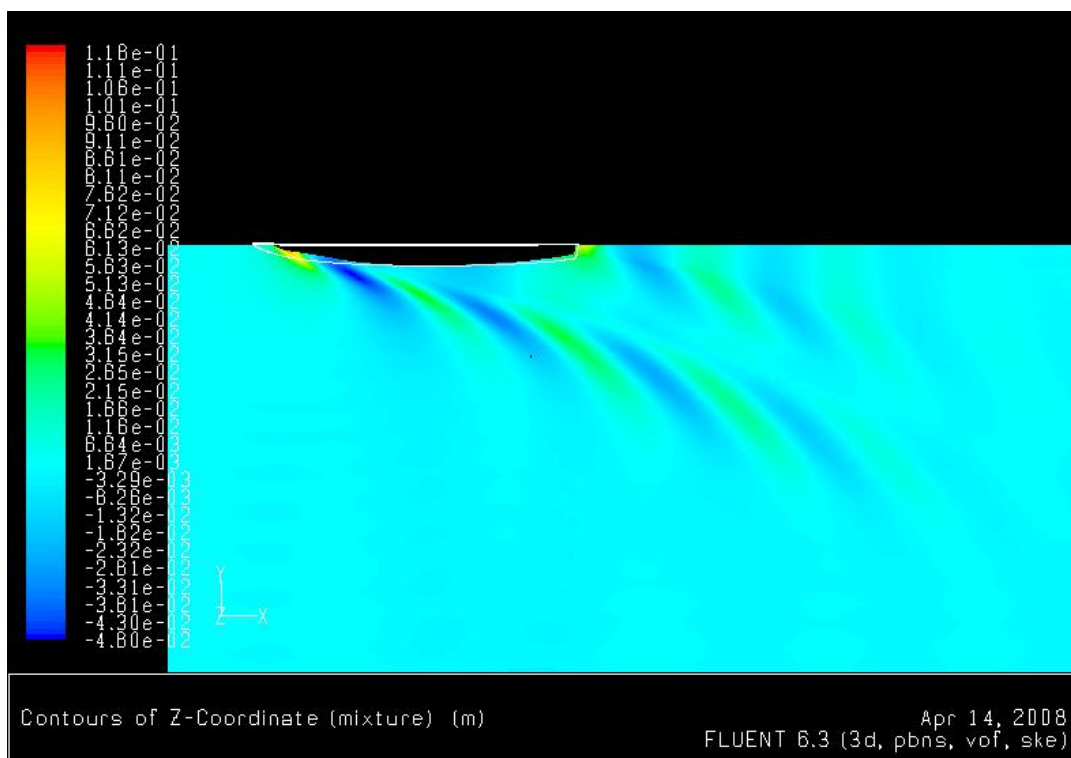
Figure 5. Wake pattern obtained after 8,000 iterations using a rectangular domain
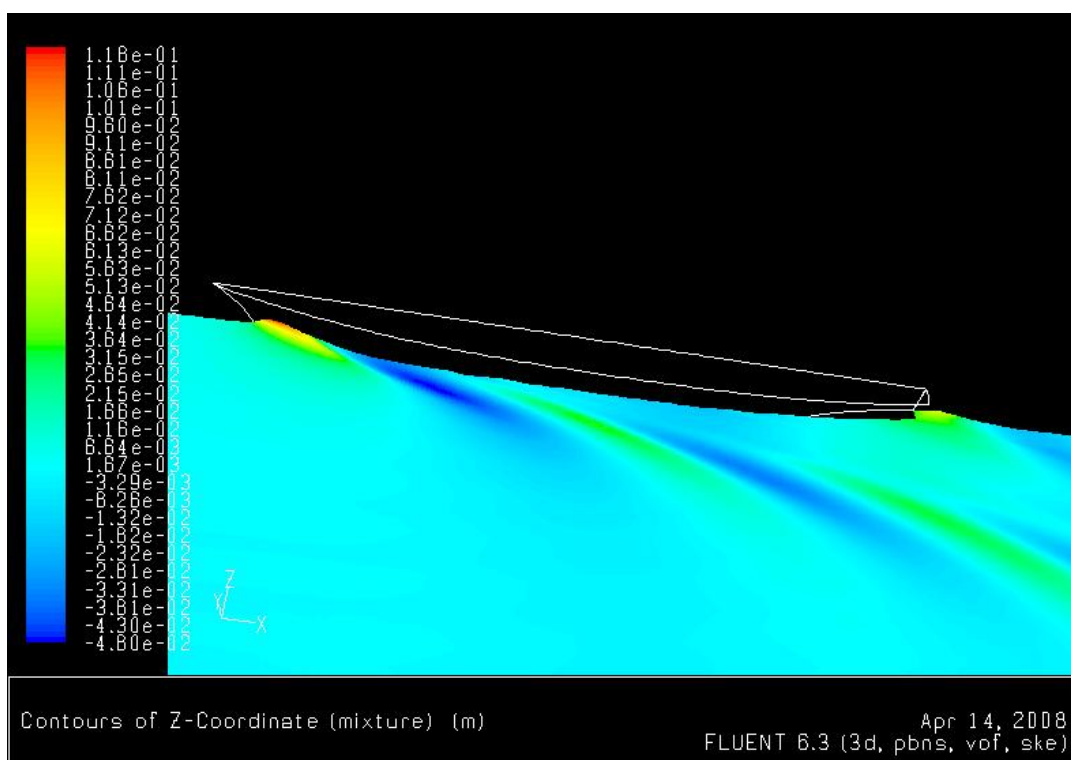

Figure 6. Detailed view of the wave pattern in the vicinity of the hull

The total resistance coefficient calculated from this simulation is the sum of the total viscous force plus the total pressure force on the hull and has the value 4.42×10^{-3}, which is 4.5% higher than the experimental value of 4.23×10^{-3} [3]. The friction coefficient has the value 3.07×10^{-3}, which is only 1.3% higher than the average result from the seven simulations reported at the Gothenburg 2000 workshop [3], while the pressure coefficient, which has a value of 1.36×10^{-3}, is 12.4% higher than the average value from the same seven simulations. Rhee and Skinner [19], who more recently performed a simulation of the DTMB hull using the Fluent code and the more sophisticated RSM turbulence model, found a total resistance coefficient of 3.83×10^{-3}, which is an under prediction of 9.4% from the experimental value.

Figure 7 shows the simulated wave profile along the hull surface compared with the experimental data from the Gothenburg test set [27]. Overall there is good qualitative agreement between the simulated and experimental profiles. The phase and amplitude of the primary bow wave are slightly in error but this is a common problem in this type of simulation. Rautaheimo and Salminen [28] under predicted the peak amplitude by 25% in their simulation of the DTMB 5415 using the FINFLOW code. The only significant disagreement with the experimental data is the strong dip in the water surface immediately following the bow wave. This is not seen in the simulations of Rautaheimo and Salminen [28], Rhee and Skinner [19] or Chen et al. [18], but is noticeable in the simulation of Li [14] using the FINFLOW-RANS SHIP solver.

Better agreement between the simulated and experimental data can be seen in Figure 8, which shows the wave profile along the line at y/Lpp = 0.172. There is excellent agreement in the shape of the profile, although the phase difference between the simulated and experimental results slowly increases along the length of the cut.

The ability to accurately simulate the wake flow in the vicinity of the propeller plane is important for the prediction of propeller loadings and Figure 9 shows a comparison between the simulated and experimental velocity components on the propeller plane at z/Lpp = -0.02. There is excellent agreement for the y (v/U) and z (w/U) components while the maximum discrepancy for the axial component (x/U) is around 10%.

To check any grid dependency of these results we adapted the grid in the region 20 cm above and below the water line, resulting in a mesh with approximately 11.5 million hexahedral cells. The simulation was then continued out to 17,000 iterations until the residuals had again decreased to acceptable levels. Visual inspection of the water surface showed effectively no change to the wave pattern and detailed comparison with the individual wave profiles and the results calculated on the coarser mesh showed that there was very little change to the simulated results.
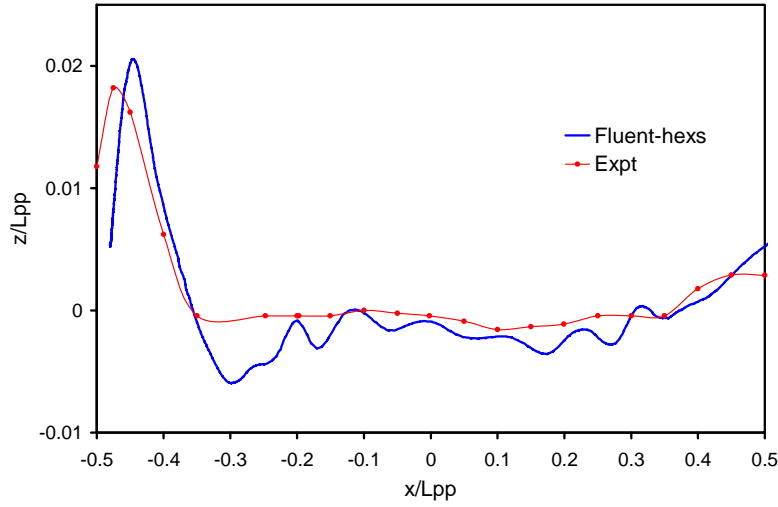
Figure 7. Comparison of simulation with experimental results: wave profile along the hull for fully structured hexahedral mesh
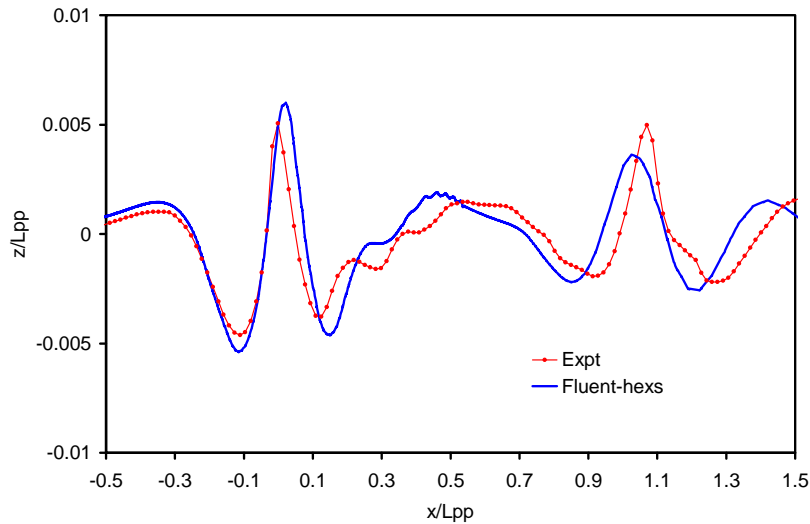

Figure 8. Comparison of simulation with experimental results: wave profile along the line at y/Lpp = 0.172 for fully structured hexahedral mesh

Plots for the wave profile along the hull and along the line at $y/\text{Lpp} = 0.172$ are shown in Figures 10 and 11 respectively. The finer mesh has resulted in a slight improvement in the phase of the peak bow wave, but the magnitude of the peak has moved slightly further from the experimental result. A similar result has occurred for the wave profile along the line at $y/\text{Lpp} = 0.172$, where the finer mesh has resulted in an increased height for the primary wave, but this has moved the calculated value further from the experimental result.

The total resistance coefficient on the unadapted mesh has the value 4.42×10^{-3}, which is 4.5% higher than the experimental value. On the adapted mesh this changes to 4.39×10^{-3}, which agrees slightly better with the experimental value of 4.23×10^{-3} [3] and is only 3.8% in error.

14

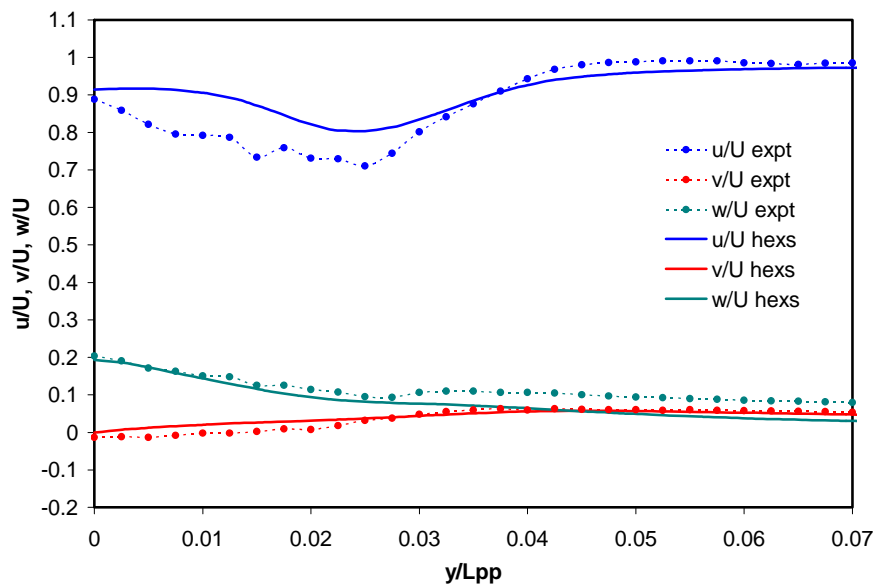*Figure 9. Comparison of simulation with experimental results: velocity components on propeller plane at z/Lpp = -0.02 for fully structured hexahedral mesh*

Figure 12 shows the effect of the grid refinement on the velocity components in the propeller plane. For the y and z components there is effectively no change to the simulated results, while for the axial component the increased grid resolution has resulted in a very slight change in the result near y/Lpp = 0.015 which brings it just slightly closer to the experimental result.
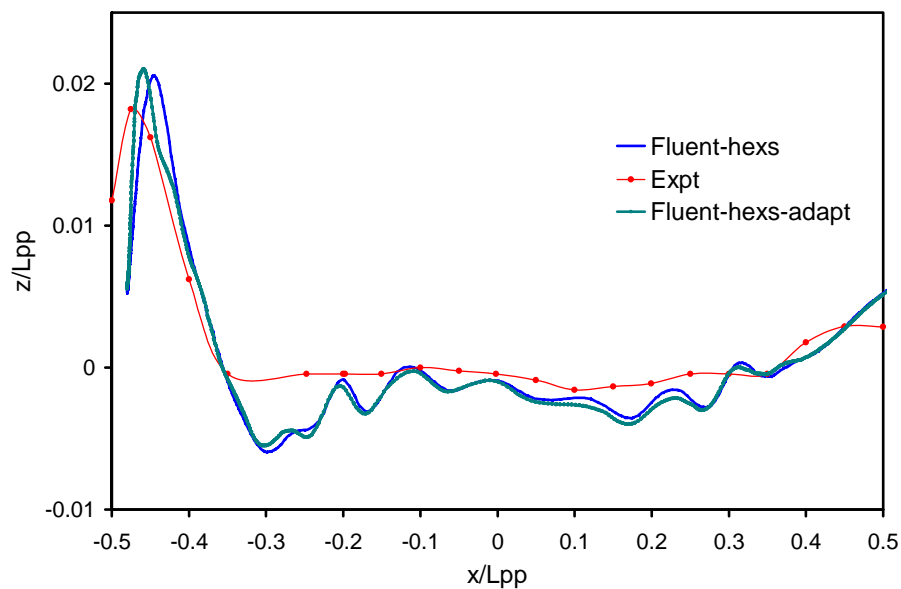

Figure 10. Effect of grid refinement: wave profile along the hull for original hexahedral mesh and refined hexahedral mesh
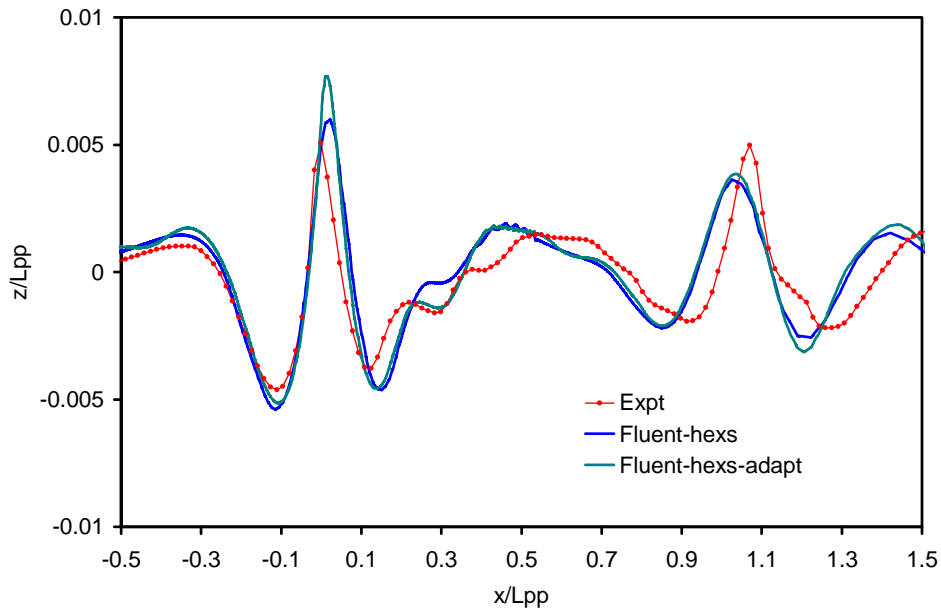
Figure 11. Effect of grid refinement: wave profile along the line at y/Lpp = 0.172 for original hexahedral mesh and refined hexahedral mesh
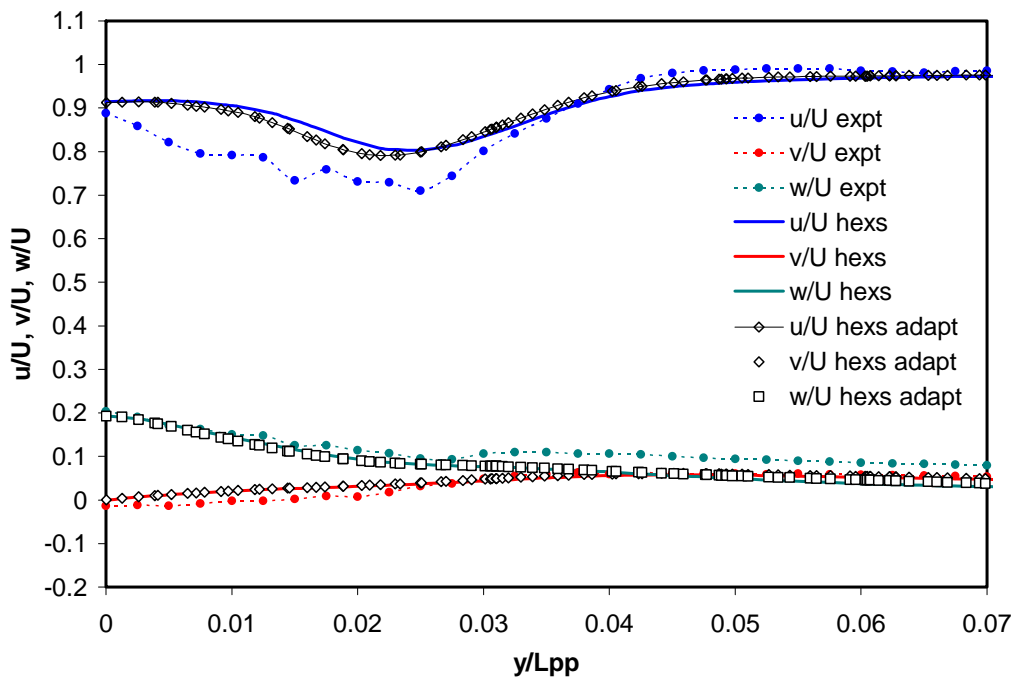

Figure 12. Effect of grid refinement: velocity components on propeller plane at z/Lpp = -0.02 for original hexahedral mesh and refined hexahedral mesh

6. Simulation Results with a Tetrahedral/Prism Mesh

One of the advantages of the Fluent code for simulations of naval vessels is its ability to solve the RANS equations on either a structured hexahedral mesh or an unstructured hybrid mesh consisting of tetrahedral cells and prism cells. It is generally accepted that hexahedral meshes provide more accurate simulation results [25], but the one disadvantage of this approach is the time required to construct the mesh. The hexahedral mesh used in the previous section took approximately five days to construct, while the hybrid mesh used in this section was constructed in one day. Whilst the unappended DTMB 5415 is a relatively simple shape to mesh using either a hexahedral or hybrid scheme this will not necessarily be the case when simulations on fully appended hulls are required. In that case, especially when using the Gambit software package to create the mesh, the use of an unstructured hybrid mesh will result in considerable time savings. In this section we have used an unstructured hybrid mesh to gain some experience with this technique in anticipation of the need to consider more complicated geometries in future naval applications.

A half domain grid was again used but the size of the domain was smaller than that used with the hexahedral mesh, in this case extending a distance 2 Lpp aft of the hull, one Lpp upstream from the bow, 1.5 Lpp out from the symmetry plane, and the total vertical depth of the domain was one Lpp. The undisturbed sea surface was located on the $z = 0$ plane. A surface mesh for the hull was constructed from triangular elements with side lengths of approximately 2cm. A Gambit curvature size function was used to ensure smooth growth of the mesh size in highly curved regions of the hull. The maximum angle between the normals to the surface was set to 20°. The hull top, undisturbed sea surface and symmetry plane were meshed with triangular elements having sizes of 5cm, 10cm and 50cm respectively with growth rates of approximately 1.15. Prism layers were then grown from the hull surface, the symmetry plane and the plane representing the undisturbed sea surface. The first cell height for each of these surfaces was set to 5.72mm. This provides good resolution for waves on the sea surface but is far too large to achieve the optimum y^{+} value of 100 on the ship surface. We therefore expect that our simulated friction drag will not be very accurate, but the wave profiles should be well resolved. A tetrahedral volume mesh was then grown from each of these surfaces, again using appropriate size functions to ensure a uniform rate of change in cell size, and the resulting mesh had 1.8 million cells.

The simulation was again run using an implicit steady-state cell based solution procedure with the SIMPLE algorithm for the pressure-velocity coupling, the PRESTO scheme for the pressure interpolation, the 2nd order upwind scheme for the solution of the momentum equations and the modified HRIC scheme for the solution of the volume fraction equations. The standard k-ε turbulence model with equilibrium wall functions was again used to simulate turbulent flow. The simulation was run for 25,000 iterations and the resulting wave pattern is shown in Figure 13.

The total resistance coefficient from this simulation has the value 3.40×10^{-3}, which is approximately 20% lower than the experimental value of 4.23×10^{-3} [3]. This is quite a good result, considering that the y^{+} value around the hull surface is approximately 1.0×10^{4}, which

is two orders of magnitude greater than the optimum value for the use of wall functions. The result is somewhat fortuitous however as the friction coefficient has the value 1.6×10^{-3}, which is approximately 40% lower than the average result from the simulations reported at the Gothenburg 2000 workshop [3]. This lower frictional drag is roughly compensated by a higher pressure drag, since the pressure coefficient has a value of 1.80×10^{-3}, which is approximately 48% higher than the average value from the simulations at the workshop.
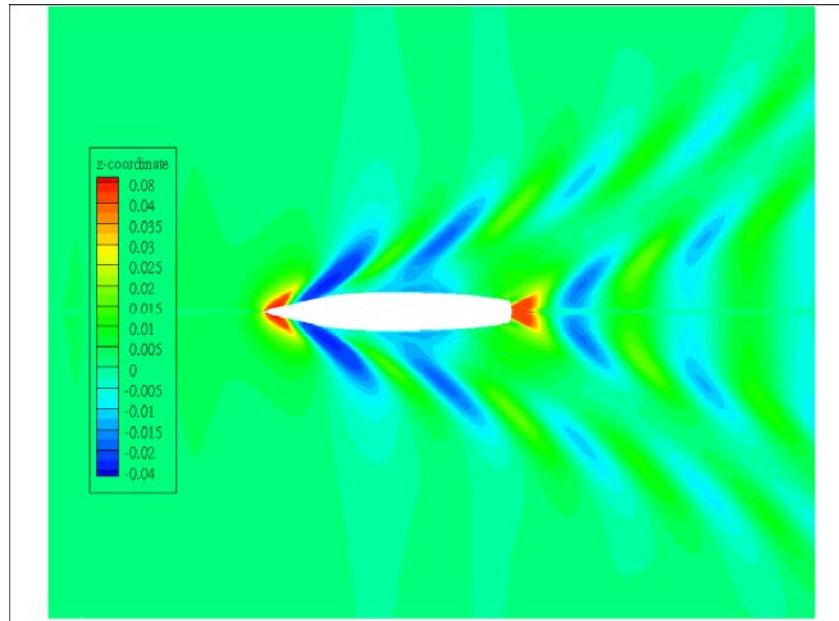

Figure 13. Wave pattern obtained after 25,000 iterations on a tetrahedral mesh.

Figure 14 shows the simulated wave profile along the hull surface compared with the experimental data from the Gothenburg test set [27] and the previous simulation results from the (unadapted) hexahedral mesh. Once again there is overall good qualitative agreement between the simulated and experimental profiles, with the degree of error from the results on the hybrid mesh being similar to those from the hexahedral mesh. The phase of the primary bow wave is accurately predicted but the amplitude is too low. This is undoubtedly due to the lower resolution of the hybrid mesh compared to the hexahedral mesh in the vicinity of the sea/air interface. The strong dip in the water surface immediately following the bow wave is still present but is not as pronounced, although this could again be due to the lower resolution of the hybrid mesh.

Figure 15 shows the wave profile along the line at $y/Lpp = 0.172$ and it can be seen that there is quite good agreement between the profiles calculated on the hybrid mesh and on the hexahedral mesh. The major difference between the two results again being evident in the calculated amplitudes, where the relative coarseness of the hybrid mesh has resulted in significantly lower peak amplitudes.

The calculated velocity components on the propeller plane at $z/Lpp = -0.02$ are shown in Figure 16. The agreement between the y component, (v/U), computed on the hybrid mesh

and on the hexahedral mesh is excellent, and both results are in good agreement with the experimental result. For the z component, (w/U), there is a slight disagreement between the results simulated on the hybrid and the hexahedral meshes quite close to the symmetry plane but the results soon become indistinguishable as y/Lpp increases. The axial component (x/U) displays quite a different shape from both the experimental data and the simulated result on the (unadapted) hexahedral mesh for y/Lpp < 0.04, but good agreement over the rest of the range.
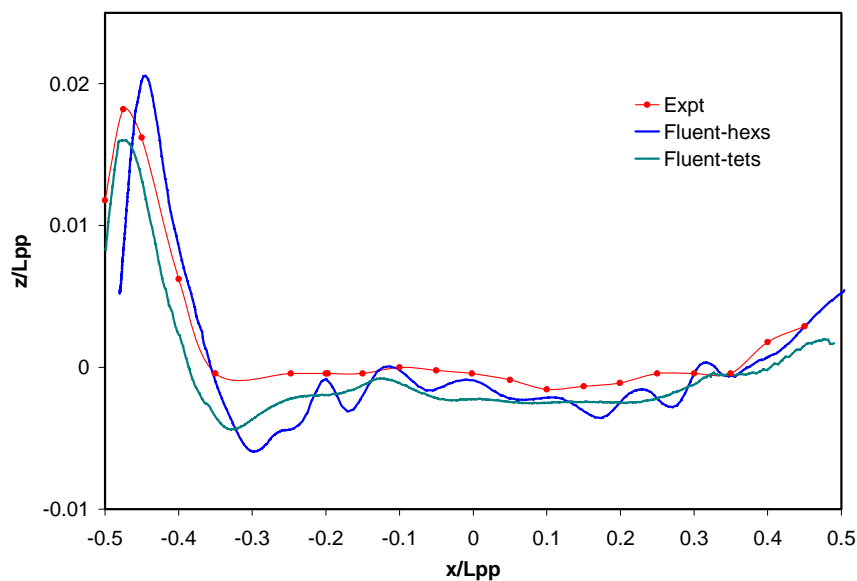
Figure 14. Comparison of simulation results computed on hexahedral and tetrahedral meshes: wave profile along the hull
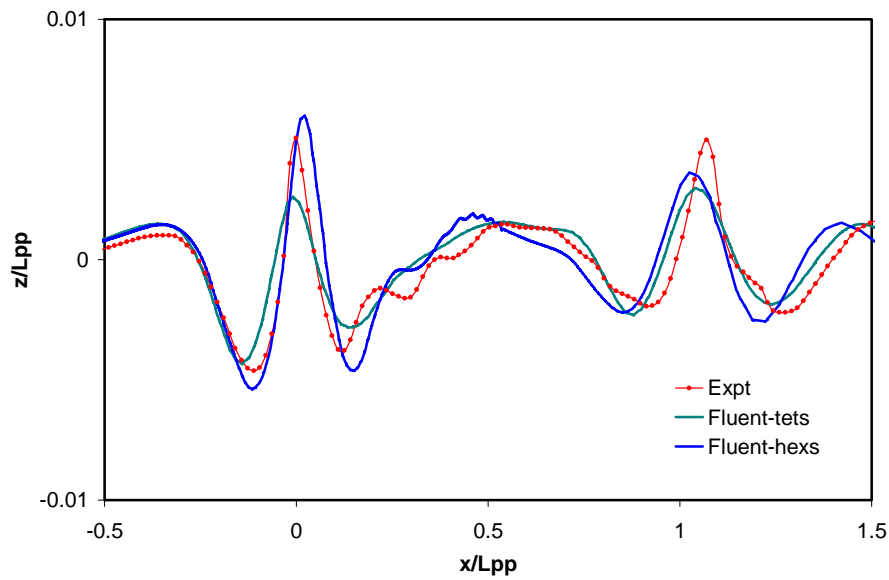
Figure 15. Comparison of simulation results computed on both hexahedral and tetrahedral meshes: wave profile along the line at y/Lpp = 0.172
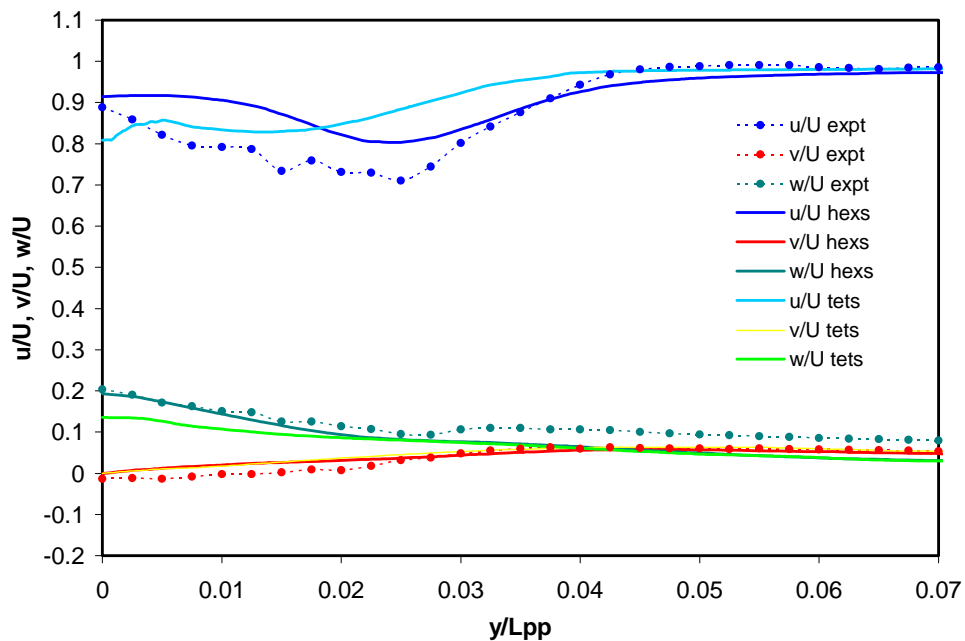

Figure 16. Comparison of simulation results computed on both hexahedral and tetrahedral meshes: velocity components on propeller plane at z/Lpp = -0.02

To check the effect of the grid on these results we again adapted the mesh in the region 20 cm above and below the water line. The adapted mesh had 3.48 million mixed cells and the simulation was then continued out to 38,500 iterations until the calculation converged. Figure 17 shows the effect of the grid refinement on the wave profile along the hull. The finer mesh

has resulted in an increase in the amplitude of the bow wave so that it now agrees with the experimental result. Apart from the increase in peak amplitude however this is essentially no difference in the profiles calculated on the two meshes.

A similar result has occurred for the wave profile along the line at y/Lpp = 0.172, shown in Figure 18. The finer mesh has resulted in an increased height for the primary and secondary waves and this has moved the calculated result closer to the experimental value. Elsewhere there is very little difference between the results calculated on the two meshes.

Figure 19 shows the effect of the grid refinement on the velocity components in the propeller plane. The results are similar to those shown in Figure 12 for the effect of mesh refinement on the hexahedral mesh. For the y and z components there is effectively no change to the simulated results, while for the axial component the increased resolution has resulted in a significant change to the computed result in the region $0.0 \leq y/Lpp \leq 0.04$, although this change has not significantly increased agreement with the experimental result.
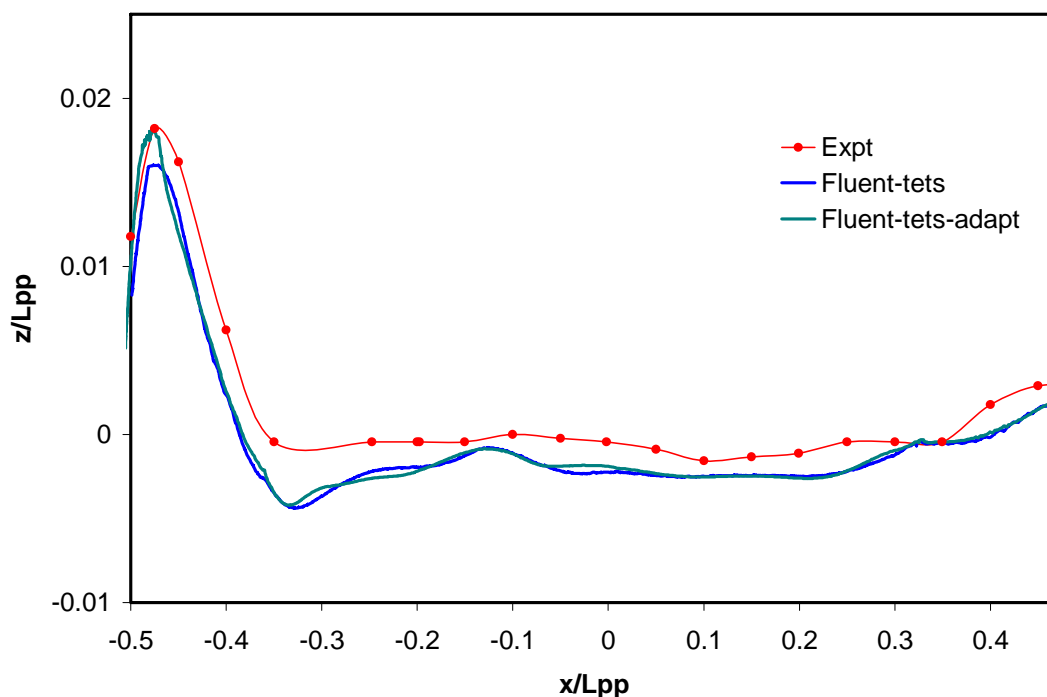

Figure 17. *Effect of grid refinement: wave profile along the hull for original tetrahedral mesh and refined tetrahedral mesh*
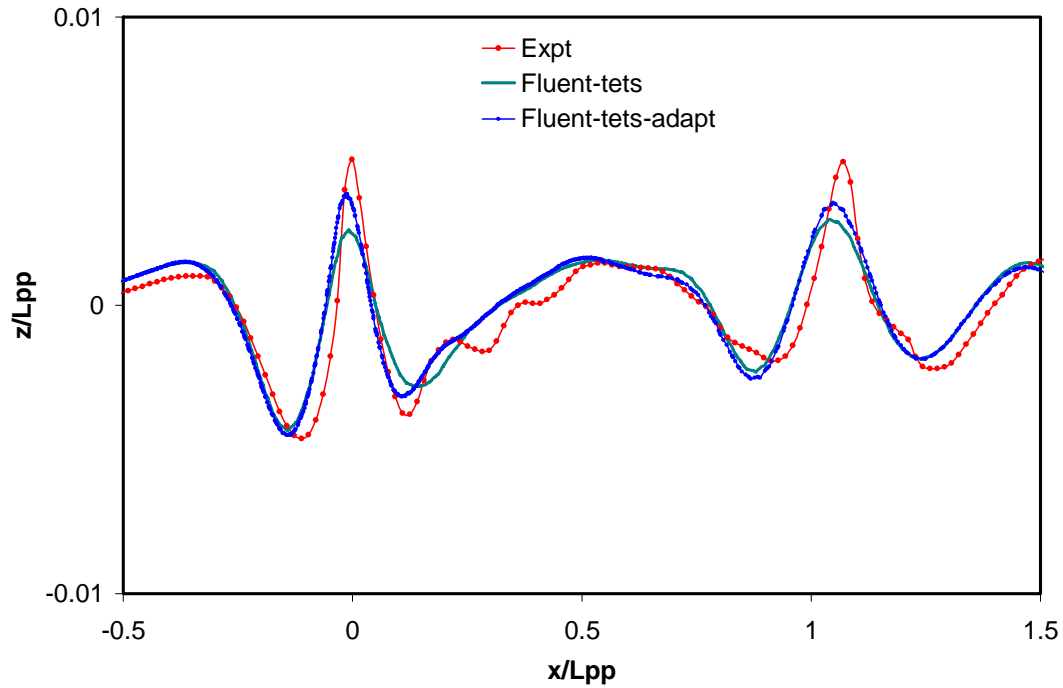
Figure 18. *Effect of grid refinement: wave profile along the line at y/Lpp = 0.172 for original tetrahedral mesh and refined tetrahedral mesh*
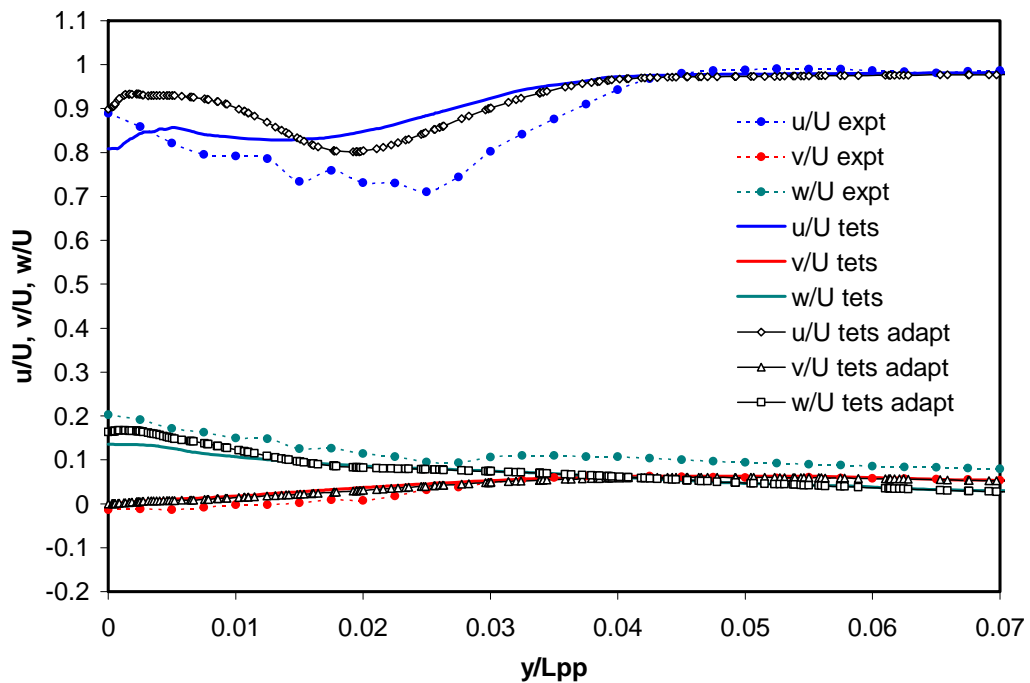

Figure 19. *Effect of grid refinement: velocity components on propeller plane at z/Lpp = -0.02 for original tetrahedral mesh and refined tetrahedral mesh*

7. Discussion

After having completed the simulation on the hexahedral mesh it was noticed that the hull had been set to an incorrect value of trim for the prescribed speed of 4.01 knots. Having set the hull to the correct sinkage at the midship point it was then rotated through an angle of 1°, rather than 0.1°, which meant that the sinkage at the Aft Point was -0.0096L and at the Forward Point it was +0.0064L. This is similar to the trim for the ship travelling at a speed of 6.0 knots rather than 4.01 knots, although the FP is higher in this case. We attempted to correct this error in the first instance by going back to the Gambit journal file and rotating the hull, and the attached mesh near to the surface of the hull (which had initially required the most work), but this led to substantial changes in the edge and face numbering scheme so that the journal file failed to run correctly. We believe that this problem stems from a degree of incompatibility between the Gambit software and the way the face geometry was described in the initial geometry file, which was obtained from the Carderock Division, Naval Surface Warfare Center [29].

Since considerable work would have been required to remesh the wireframe model from scratch at the correct values of sinkage and trim we decided to overcome this problem by rotating the entire mesh relative to the coordinate system (and therefore the inflow boundary condition) so that the model was placed at the correct sinkage and trim. This meant that the model was now at the correct attitude with respect to the flow, but this approach created several new problems. Because of the large extent of the domain, and the relatively small depth of the air layer over the sea surface, when the entire mesh was rotated it meant that the outflow boundary was filled entirely with only one phase (water), rather than the correct ratio of the two phases. This situation was not allowed by the Open Channel boundary condition. The problem was easily solved by significantly increasing the height of the air layer until the correct ratio of the two phases was again approached. Another problem which emerged from this approach was that the hexahedral cells along the sea surface were now no longer aligned with the flow. Although the degree of misalignment was small, the effect on some of the high aspect ratio cells was sufficient to cause considerable problems with the smooth running of the simulation. The relaxation factors had to be considerably reduced for example in order to avoid frequent code crashes.

The main problem encountered with this approach however was that the air/sea interface became considerably disrupted during the course of the simulation. Large amounts of water were found to be suspended in the air in completely irregular patterns. A remedy for this problem was eventually found after discussions with a Fluent engineer [30] and consisted of "repatching" the appropriate part of the domain with the air phase during the course of the simulation. This "ad-hoc" fix worked well on a Fluent machine, but was not 100% successful for our simulation, which was run on an eight processor Beowulf cluster using Opteron processors. In particular, the problem was found to persist on the symmetry plane no matter how many times we "repatched" the air domain. However, when the simulation was moved to a quad core Intel processor this problem did not occur, although problems still persisted on the symmetry plane.

Although we continued this simulation for a considerable length of time the sea surface never achieved the correct wave pattern. Figure 20 shows that the disturbances on the symmetry plane lead to the formation of a non-physical wave some distance ahead of the bow of the ship. This removed energy from the real leading bow wave, which was then considerably reduced in amplitude. The reason for this unphysical wave some distance in front of the bow can be seen in Figure 21, which shows the air/sea interface in the region forward of the bow, on which is superimposed lines representing the partitioning of the domain amongst some of the eight processors. The occurrence of the unphysical wave at the interface between the two partition regions leads us to believe that there may be some problems with the parallel version of the Fluent software. It should however be pointed out that this problem did not occur when the same simulation was run by a Fluent engineer on a different computer system. The reasons for these discrepancies are still under investigation by Fluent.

Given that the simulation results presented in Section 5 were run at an incorrect value of trim it is important to consider any effect this may have on the validity of the results. Yang and Löhner [31] have used an unstructured grid-based, parallel free-surface flow solver which has been extended to simulate sinkage and trim effects for the calculation of steady ship waves. They applied their code to the simulation of wave profiles, sinkage and trim, and wave drag for two hull forms. Their results show that while achieving the correct sinkage and trim is important for accurate predictions of wave drag, the effect on the computed wave profiles is miniscule. Given their results, we believe that the wave profiles calculated in Section 5 would not be significantly different if the simulation were to be repeated using the correct value for the trim.
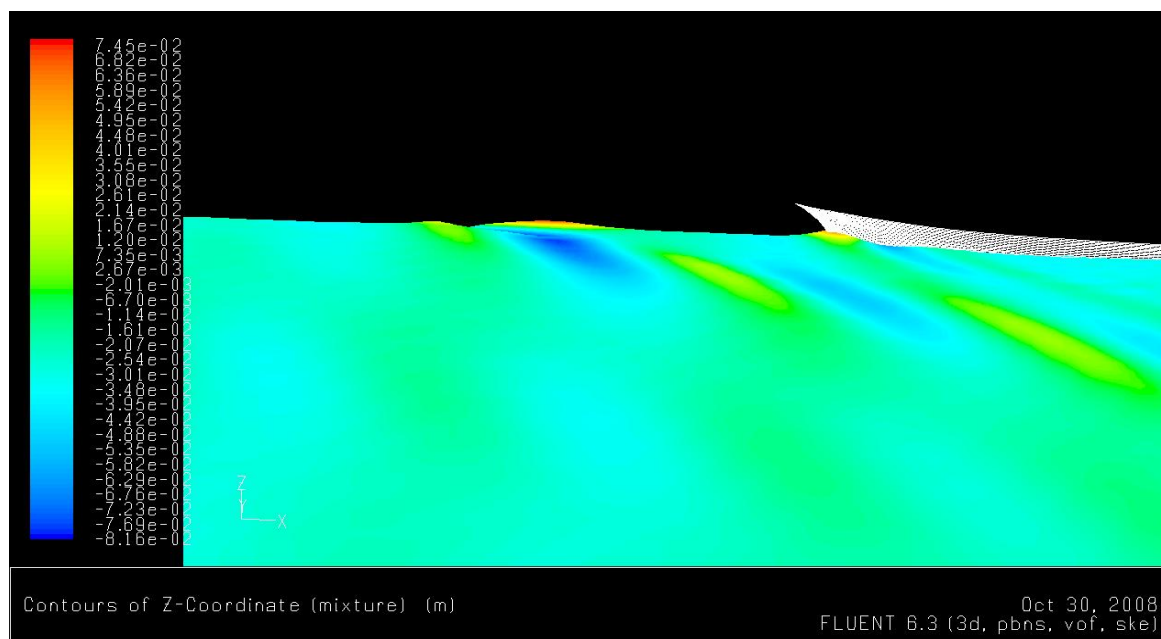

Figure 20. Air/sea interface showing the unphysical wave at some distance forward of the bow
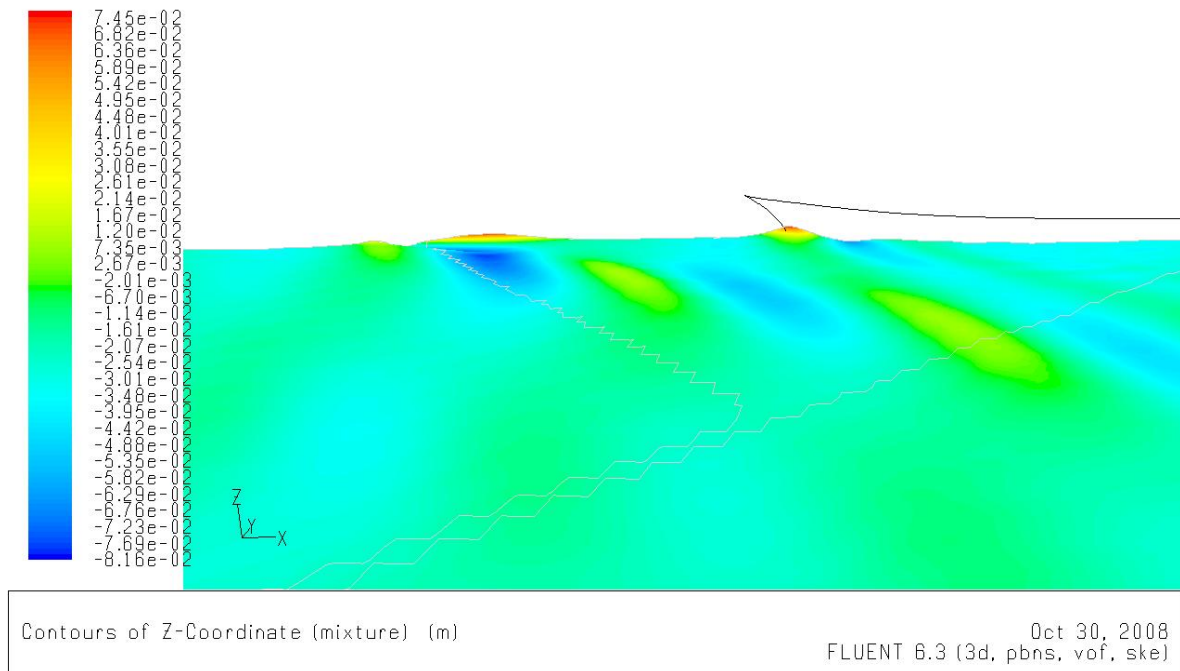
Figure 21. *Air/sea interface showing the unphysical wave at some distance forward of the bow with superimposed lines representing the partitioning of the domain amongst some of the eight processors*

8. Conclusion

We have used Fluent to calculate the free-surface flow around the DTMB 5415 naval ship model. Simulations were performed using both a structured hexahedral mesh and an unstructured tetrahedral mesh of lower resolution. Our results show that Fluent is able to accurately simulate the total ship resistance, near-field wave shapes, and the velocity field in the propeller plane. The total resistance coefficient calculated on the hexahedral mesh was found to agree with the experimental value to within 3.8%. The agreement using the tetrahedral mesh was considerably less accurate, however this was to be expected due to the low resolution of this mesh in the boundary layer region. The simulated wave shapes along the surface of the hull and in the near field computed on both the hexahedral and tetrahedral meshes were found to be in good qualitative agreement between the simulated and experimental profiles, with the degree of error from the results on the tetrahedral mesh being similar to those from the hexahedral mesh. Simulations of the axial velocity component in the propeller plane showed errors of approximately 10% on the original hexahedral mesh and this was not significantly improved when the simulation was run on the adapted mesh. Good agreement between the calculated and experimental results was found for the other velocity components. Similar results were found for simulations on the tetrahedral mesh.

The above results are encouraging and indicate that the Fluent code is a viable tool which could be considered for use in more demanding naval problems, such as the computation of

ship wakes undergoing specified manoeuvres, for example. Use of the code is far from routine however and a number of significant problems were encountered during the course of this work. They are summarized here to facilitate future applications of this code to similar naval problems:

- The Open Channel boundary condition provides an efficient approach to the performance of these simulations, but it appears to be incompatible with either a curved domain boundary surface or with an irregular surface mesh over the inlet boundary. Simulations using the Open Channel boundary condition with a regularly mapped rectangular mesh at the inlet, which transitioned downstream to a less ideal mesh geometry via a non-conformal interface, were successful.

- We experienced problems calculating an accurate sea surface shape when using a Beowulf cluster based on the Opteron processors. This problem did not occur on a SMP machine using Xeon processors. The source of this problem was not identified.

- Non-physical behaviour was observed on the symmetry plane using both Opteron and Intel processors. These problems can obviously always be avoided by removing the symmetry plane from the simulation and integrating over the full domain. This will result in an increase in computational time and resources however, so further effort should be made to discover the source of this non-physical behaviour.

- Fluent appears to be particularly sensitive to the angle of the grid with respect to the free surface water line in simulations of this type. This is far from ideal, but the problem can be avoided by remeshing the geometry for each individual sinkage and trim calculation.

9. References

1. Larsson, L., Patel, V.C. and Dyne, G. , (Eds.) "Ship Viscous Flow - Proceedings of 1990 SSPA-CTH-IIHR Workshop", Flowtech International Research Report, No 2, Gothenburg, Sweden, 1991.

2. Kodama, Y., Takeshi, H., Hinatsu, M., Hino, T., Uto, S., Hirata, N. and Murashige, S., "Proceedings of the 1994 CFD Workshop", Ship Research Institute, Japan, 1994.

3. Larsson, L., Stern, F. and Bertram, V., "Benchmarking of Computational Fluid Dynamics for Ship Flows: The Gothenburg 2000 Workshop", *Journal of Ship Research*, **47**, No.1, pp. 63-81, March 2003.

4. Hino, T. (ed.), "Proceedings of the CFD Workshop Tokyo 2005", Tokyo, Japan, 2005.

5. Burg, C.O.E, Sreenivas, K., Hyams, D.G. and Mitchell, B., "Unstructured Nonlinear Free Surface Simulations for the Fully-Appended DTMB Model 5415 Series Hull Including Rotating Propulsors", *Proceedings of the 24th Symposium on Naval Hydrodynamics*, Fukuoka, Japan, 8-13 July, 2002.

6. Wilson, W., Fu, T.C., Fullarton, A. and Gorski, J., "The Measured and Predicted Wave Field of Model 5365: An Evaluation of Current CFD Capability", presented at the *26th Symposium on Naval Hydrodynamics*, Rome, Italy, 17-22 September, 2006.

7. Gregory, P., " Computational Fluid Dynamics Analysis of the Flow over a Collins Class Submarine", DSTO-CR-2005-0144 (Confidential), 2005.

8. Gregory, P., "Evaluation of hull modifications for the Collins Class Submarine using Computational Fluid Dynamics", DSTO-CR-2005-0145 (Confidential), 2005.

9. Gregory, P.A. and Chen, L., "Computational Fluid Dynamics Analysis of Flow Disturbances Created by Fin-mounted Cameras on a Collins Class Submarine", DSTO *Technical Report*, DSTO-TR-1649, (Restricted), February 2005.

10. Widjaja, R., Anderson, B., Chen, L. and Ooi, A. "RANS Simulation of Suboff Bare Hull Model", DSTO-CR-2006-0463, January 2007.

11. Jones, D.A. and Clarke, D.B., "Simulation of a Wing-Body Junction Experiment using the Fluent Code", DSTO *Technical Report*, DSTO-TR-1731, June 2005.

12. Jones, D.A. and Clarke, D.B. "Simulation of Flow Past a Sphere using the Fluent Code", DSTO *Technical Report*, DSTO-TR-2232, 29pp, December 2008.

13. Burg, C.O. and Marcum, D.L., "Moving Towards High-Fidelity RANS Calculations Of Maneuvering Surface Vessels Using Unstructured Grids", *8th International Conference on Numerical Ship Hydrodynamics*, Busan, Korea, Sept. 22-25, 2003.

14. Li, T., "Computations of turbulent free-surface flows around modern ships", *Int. J. Numer. Meth. Fluids*, **43**, 407-430 (2003).

15. Hirt, C.W. and Nichols, B.D., "Volume-of-fluid (VOF) method for dynamics of free boundaries", *J. Comput. Phys.*, **39**, 201-221 (1981).

16. Osher, S. and Sethian, J.A., "Fronts propagating with curvature-dependant speed: algorithms based on Hamilton-Jacobi formulations.", *J. Comput. Phys.*, **79**, 12-40 (1988).

17. Sussman, M., Smekerda, P. and Osher, S.J., "A level set approach for computing solutions to incompressible two-phase flow", *J. Comput. Phys.*, **114**, 146-159 (1994).

18. Chen, Y-J, Chau, S-W and Kouh, J-S., "Application of two-phase fluid approach for free surface ship flow simulation", *Journal of the Chinese Institute of Engineers*, **25**, No.2, 179-188 (2002).

19. Rhee, S. and Skinner, C., "Unstructured Grid Based Navier-Stokes Solver for Free-Surface Flow around Surface Ships", In: *Proceedings of the workshop on CFD ship hydrodynamics*, pp. 504-509, March 9-11, Tokyo, Japan, 2005.

20. Yang, J., Sakamoto, N. and Wang, Z., "Two Phase Level-Set/Immersed Boundary Cartesian Grid Method for Ship Hydrodynamics", presented at the *9th International Conference on Ship Hydrodynamics*, Ann Arbor, Michigan, August 5-8, 2007.

21. Di Mascio, A., Broglia, R. and Muscari, R., "On the application of the single-phase level set method to naval hydrodynamic flows", *Computers and Fluids*, **36**, 868-886 (2007)

22. Carrica, P.M., Wilson, R.V., Noack, R.W and Stern, F., "Ship motions using single-phase level set with dynamic overset grids", *Computers and Fluids*, **36**, 1415-1433 (2007).

23. Stern, F., Longo, L., Penna, R., Oliviera, A., Ratcliffe, T. and Coleman, H., "International Collaboration on Benchmark CFD Validation Data for Surface Combatant DTMB Model 5415", *23rd Symposium on Naval Hydrodynamics*, Val de Reuil, France, 17-22 September 2000.

24. Gorski, J.J., "Marine Vortices and Their Computation", paper presented at the RTO AVT Symposium on "*Advanced Flow Management: Part A - Vortex Flows and High Angle of Attack for Military Vehicles*", held in Loen, Norway, published in RTO-MP-069(1), 7-11 May 2001.

25. Wood, M.P., González, L.M., Izquierdo, J., Sarasquete, A. and Rojas, L.P., "RANSE with free surface computations around fixed DTMB 5415 model and other Baliño's fishing vessels", presented at the 9th International Conference on Numerical Ship Hydrodynamics, Michigan, USA, 5-8 August, 2007.

26. Muzaferija, S., Peric, M., Sames, P. and Schellin, T., "A Two-Fluid Navier-Stokes Solver to Simulate Water Entry". In *Proc 22nd Symposium on Naval Hydrodynamics*, pages 277-289, Washington, DC, 1998.

27. http://www.iihr.uiowa.edu/gothenburg2000/5415/combatant.html

28. Rautaheimo, P and Salminen, E., "Simulation of a free-surface flow", Helsinki University of Technology CFD-group/ Laboratory of Applied Thermodynamics *Memo No. CFD/TERMO-32-2000*, April 2000.

29. e-mail communication from Toby Ratcliffe, Naval Surface Warfare Center, Carderock Division, containing an iges file of the bare hull of Model DTMB 5415, 22nd November, 2007.

30. e-mail communication from Hawk Lee, LEAP Australia Pty Ltd, 8th August, 2008.

31. Yang, C., and Löhner, R., "Calculation of Ship Sinkage and Trim Using a Finite Element Method and Unstructured Grids", *International Journal of Computational Fluid Dynamics*, **16**, 217-227 (2002).

DEFENCE SCIENCE AND TECHNOLOGY ORGANISATION DOCUMENT CONTROL DATA		1. PRIVACY MARKING/CAVEAT (OF DOCUMENT)

2. TITLE Fluent Code Simulation of Flow around a Naval Hull: the DTMB 5415	3. SECURITY CLASSIFICATION (FOR UNCLASSIFIED REPORTS THAT ARE LIMITED RELEASE USE (L) NEXT TO DOCUMENT CLASSIFICATION) Document (U) Title (U) Abstract (U)

4. AUTHOR(S) D.A. Jones and D.B. Clarke	5. CORPORATE AUTHOR DSTO Defence Science and Technology Organisation 506 Lorimer St Fishermans Bend Victoria 3207 Australia

6a. DSTO NUMBER DSTO-TR-2465	6b. AR NUMBER AR-014-843	6c. TYPE OF REPORT Technical Report	7. DOCUMENT DATE September 2010

8. FILE NUMBER 2009/1136160	9. TASK NUMBER NAV 07/055	10. TASK SPONSOR DNPS	11. NO. OF PAGES 30	12. NO. OF REFERENCES 31

13. DOWNGRADING/DELIMITING INSTRUCTIONS To be reviewed three years after date of publication DSTO-TR-2465	14. RELEASE AUTHORITY Chief, Maritime Platforms Division

15. SECONDARY RELEASE STATEMENT OF THIS DOCUMENT *Approved for public release* OVERSEAS ENQUIRIES OUTSIDE STATED LIMITATIONS SHOULD BE REFERRED THROUGH DOCUMENT EXCHANGE, PO BOX 1500, EDINBURGH, SA 5111

16. DELIBERATE ANNOUNCEMENT No Limitations

17. CITATION IN OTHER DOCUMENTS   Yes

18. DSTO RESEARCH LIBRARY THESAURUS http://web-vic.dsto.defence.gov.au/workareas/library/resources/dsto_thesaurus.shtml Computational Fluid Dynamics, Navier-Stokes equations Wakes Naval Ships

19. ABSTRACT This report describes the application of the Fluent code to the numerical simulation of the free-surface flow around a model naval ship; the DTMB 5415. Simulations were performed using both a structured hexahedral mesh and an unstructured tetrahedral mesh of lower resolution. The results show that Fluent is able to accurately simulate the total ship resistance, near-field wave shapes, and the velocity field in the propeller plane. The results indicate that Fluent is a viable tool which could be considered for use in more demanding naval problems, such as the computation of ship wakes undergoing specified manoeuvres. A number of significant problems were encountered during the course of the work and have been summarized to facilitate future applications of the code to similar naval problems.